Numerical Limitations of Hydraulic Models

L. Toombes[1] and H. Chanson[2]
[1]Aurecon Australia Pty Ltd
32 Turbot Street Brisbane QLD 4004
AUSTRALIA
[2]The University of Queensland, School of Civil Engineering
Brisbane QLD 4072
AUSTRALIA
Email: ToombesL@ap.aurecongroup.com

Abstract: *Fluid motion is controlled by the basic principles of conservation of mass, energy and momentum, which form the basis of fluid mechanics and hydraulic engineering. Complex flow situations must be solved using empirical approximations and numerical models, which are based on derivations of the basic principles (backwater equation, Navier-Stokes equation etc). All numerical models are required to make some form of approximation to solve these principles, and consequently all have their limitations. Sadly, these limitations are usually neither advertised by the software developers, nor investigated and understood by the users. The consequences of misusing a model can be catastrophic. This paper presents a brief explanation of the backwater and Navier-Stokes equations, and examines the use of these equations in the context of several leading software packages. The major assumptions and approximations implicit in both the base equations and numerical models are identified, together with the limitations that these impose. The validity of the numerical models is then examined in a number of situations, including verification of the model predictions against physical model data.*

Keywords: *Basic equations, numerical modelling, physical modelling, limitations*

1. INTRODUCTION

Fluid motion is controlled by three basic principles: conservation of mass, energy and momentum. Derivatives of these principles are commonly known as the continuity, energy and momentum equations. These principles are among the first taught in basic fluid mechanics, and they form the foundation of the field of hydraulic engineering. However, as situations become increasingly complex, we lose track of these essential principles. Basic equations are replaced by empirical approximations, and mathematical calculations with numerical models. These are an essential part of a professional engineer's life. Determining an equivalent surface roughness of a floodplain is far more difficult than estimating an equivalent roughness height or a Manning's roughness coefficient; solving a backwater equation for an irregular channel would be an arduous task without the assistance of a numerical model. While the advantages of numerical models cannot be ignored, we run the risk of becoming mindless automatons, plugging raw data into our numerical models and blindly accepting the results that they produce.

Numerical models come in a wide range of shapes and flavours – one, two or three dimensions, steady or unsteady flow conditions etc. All are based on derivations of the basic principles. All are required to make some form of numerical approximation to solve these principles. All have their limitations. Unfortunately, software developers are usually reluctant to advertise the limitations of their products. Engineers are often remiss in their duties to understand the capability of the software they use and to validate the results they produce. The use of a model in a manner for which it was not designed, or that contravenes the approximations upon which it was based, can lead to gross errors in the model predictions. The consequences may lie anywhere between negligible and catastrophic, potentially leading to property damage and loss of life. The objective of this paper is to promote a basic awareness of how numerical models operate and to draw attention to some of the more common limitations that are implicit to this operation, in the hope that this may encourage these models to be used in (and only in) the manner for which they are intended.

2. FLUID MECHANICS 101

Fluid mechanics is the study of fluids at rest (statics) or in motion (dynamics), including the interaction between the fluid and its surroundings. As a branch of mechanics, fluid flow is governed by well known and understood basic principles. However, application of these principles is far easier in theory than in practice due to the complexity of fluid flow, including both the geometry and the properties of the fluid itself (viscosity, compressibility, surface tension etc.). It is therefore necessary to make assumptions that simplify the application of the controlling equations, and use numerical modelling techniques to obtain solutions of complex problems. The principles of fluid mechanics can be found in many textbooks (e.g. Henderson 1966, Liggett 1994), and it is not the intention of this paper to cover them in any great detail. Simple descriptions of the basic concepts and some of the more common simplifications, necessary for understanding how these principles are used by numerical modelling software, are nevertheless provided in the sections below.

2.1. Basic Principles of Fluid Mechanics

The mechanics of fluid flow is governed by three basic principles of conservation:

Mass – The Lomonosov-Lavoisier law states that the mass of a closed system (a system into which there is no inflow or outflow) remains constant, regardless of the processes acting inside the system. An equivalent statement is that matter cannot be created or destroyed, although it may be rearranged. If a system is open then the rate of increase in the mass within the control volume is equal to the cumulative mass flowrate into the control volume:

$$\int_S \dot{m}\,\mathrm{d}A = \frac{\mathrm{d}M}{\mathrm{d}t} \tag{1}$$

where $\dot{m}$ is the mass flowrate across the control surface, M is the mass within the control volume and t is time. If the fluid is incompressible (constant density) then the equation can be simplified by replacing mass with volume. For steady-state conditions, this further simplifies to Q = constant.

Momentum – Newton's second law states that the rate of change of momentum of a body is proportional to the resultant force acting on the body and is in the same direction. For a control volume, this may be written in differential vector format as:

$$\frac{\mathrm{d}}{\mathrm{d}t}\left(m \times \vec{V}\right) = \sum \vec{F} \tag{2}$$

where $\vec{F}$ is the force (vector) acting on the control volume and $\vec{V}$ is the velocity (vector) of the control volume. For a Newtonian fluid and assuming constant density (ρ) and viscosity (μ), the equation of motion may be written in the x-direction as:

$$\underbrace{\rho\left(\frac{\partial V_x}{\partial t} + V_x\frac{\partial V_x}{\partial x} + V_y\frac{\partial V_x}{\partial y} + V_z\frac{\partial V_x}{\partial z}\right)}_{Acceleration} = \underbrace{\rho g_x}_{Gravity} - \underbrace{\frac{\partial P}{\partial x}}_{Pressure} + \underbrace{\mu\left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_x}{\partial y^2} + \frac{\partial^2 V_x}{\partial z^2}\right)}_{Shear} \tag{3}$$

where V_x, V_y and V_z are the velocity components in the x, y and z directions, P is the pressure and g_x is the resultant of the gravitational acceleration (or other volume forces) in the x-direction. Similar equations derived for the y and z directions are collectively known as the *Navier-Stokes* equations.

Energy – The first law of thermodynamics states that the net energy supplied to a system is equal to the increase in energy of the system and the energy that leaves the system as work is done:

$$\frac{\mathrm{d}E}{\mathrm{d}t} = \frac{\mathrm{d}Q_h}{\mathrm{d}t} - \frac{\mathrm{d}W}{\mathrm{d}t} \tag{4}$$

where E is the energy, Q_h is the heat added to the system and W is the work done by the system. The energy of the system is the sum of the potential (gz), kinetic ($V^2/2$) and internal energy. The 'Backwater Equation' is a derivative of the Energy Equation for steady flow along a streamline:

$$\frac{\partial H}{\partial s} = -S_f = -f\frac{1}{D_H}\frac{V^2}{2g} \tag{5}$$

where H is the total head (often simplified as $H = P/\rho g + z + V^2/2g$), s is the distance along a streamline, S_f is the friction slope, f is the Darcy friction factor, and D_H is the hydraulic diameter.

2.2. Numerical Models

Computational fluid dynamics (CFD) can be defined as a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems involving fluid flows. The term "CFD model" is commonly used to refer to a high-order numerical model capable of solving complex flow situations with relatively few simplifications (eg three-dimensional, multi-fluid, compressible, thermodynamic effects etc.). In reality, all numerical models are CFD models (even a simple spreadsheet solution of the backwater equation). There are generally considered to be two methods of analysing fluid motion: by describing the detailed flow pattern at every point in the flow field (small scale or differential analysis), or by examining a finite region and determining the gross effects of and on the region (finite or control-volume analysis). Since they are generally concerned with describing or determining the fluid properties within space, most numerical models adopt a control-volume approach.

The complexity of real fluid flow makes it impossible to solve the governing equations without making some form of simplifying approximation, even with the use of complex models and fast computers. Common practices include: (a) simplification of the spatial and geometric properties (e.g. solution of the flow field in only one or two dimensions, assumption of cross-section average or depth-average properties), (b) assumption of steady or quasi-steady flow conditions (independent of time), (c) neglect of fluid properties that would have negligible influence in the circumstances being investigated (e.g. constant density and temperature, no viscosity or surface tension), and (d) use of empirical formulae to approximate flow characteristics (e.g. Manning's equation, k-ε turbulence model).

Hydraulic models may be categorized by the spatial and temporal simplifications that the model employs. Each category has associated with it a number of fluid property and dynamic assumptions (although there are always exceptions to the rule). The following sections aim to outline some of the more common categories, the assumptions typically associated with these categories, and the limitations that these impose.

3. ONE-DIMENSIONAL MODELLING

As the name implies, one-dimensional models assume that the flow is in one direction only, and there is no direct modelling of changes in flow distribution, cross-section shape, flow direction, or other two- and three-dimensional properties of the flow. The channel geometry is typically represented as a series of cross-sections at fixed (but not necessarily uniform) intervals. Although often considered to be relatively simplistic by modern standards, one-dimensional modelling remains a useful and valid tool in many situations. One-dimensional hydraulic models may be categorized as steady or unsteady. While these appear, superficially, to be similar and share many of the same limitations, the basic hydraulic principles to numerically solve these two situations are very different. Steady-state numerical models are in most cases based on a derivative of the 'backwater' (or Energy) equation, while unsteady models are based on a derivative of the Saint Venant (or Momentum) equation. Each solution has its advantages and disadvantages, and neither is appropriate in all situations. The derivation and implicit limitations of these solutions are described in the sections below.

3.1. Generic Assumptions Common to One-Dimensional Modelling

One-dimensional models make a number of approximations in line with their simplistic nature. Some are so obvious that they (hopefully) cannot be missed, while others are not so well recognised. Flow properties must be calculated based on characteristic properties of the cross-section (eg hydraulic diameter, average velocity). Some software packages try to provide greater flexibility by dividing each cross-section into sub-areas (such as the main channel, left and right overbanks), then applying various weighting factors to the flow distribution between the sub-areas and the travel distance of each component. More complex software packages can simulate quasi-2D situations as a series of inter-linked channels, however the definition of flow path and length is inflexible. Even with these abilities, one-dimensional modeling is only appropriate for modelling well-defined and constant flowpaths; the model cannot match the flexibility of two- and three-dimensional modelling necessary for representing complex channel/floodplain interactions.

A less obvious simplification common to many numerical models (eg HEC-RAS, MIKE 11) is to assume that the grade of the channel is small, nominally less than 1:10, and therefore the sine and cosine of the channel slope can be assumed equal to zero and unity respectively, allowing the cos θ term to be neglected from the calculation of hydrostatic pressure ($P = \rho g d \cos\theta$) and elevation ($z = z_0 + d \cos\theta$) shown in Figure 1. This also allows numerous geometric implications may be ignored; was the cross-section originally defined vertically or perpendicular to the invert, and are the water level results projected vertically ($d/\cos\theta$) or perpendicular to the invert ($d \cos\theta$)?

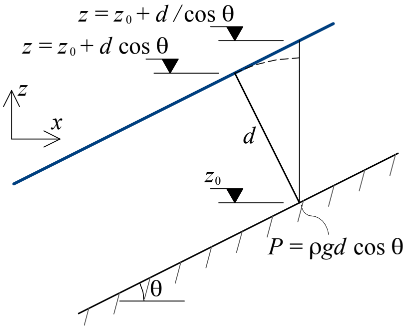

Figure 1 Level/Depth Definitions

Rather than using a physically derived coefficient, such as the Darcy friction factor f, most numerical models estimate friction losses using an empirical approximation such as Manning's coefficient. While Manning's equation has been in use for over 120 years, it is perhaps this universal acceptance that has lead to evident ignorance about the limitations of the equation. A popular misconception is that Manning's roughness coefficient is a dimensionless constant, whereas in reality it has units (s/m$^{1/3}$) and is dependent upon the hydraulic radius. Care must therefore be taken, not only in the estimation of appropriate roughness coefficients, but to realise that although a model has been calibrated for one particular discharge, the performance may be different for other flow conditions. Additionally, because there is no direct modelling of two- and three-dimensional flow effects, the roughness coefficient must account for the contribution of these aspects to hydraulic losses in the channel.

3.2. The Backwater Equation and Steady-State One-Dimensional Modelling

Numerical models usually solve the backwater equation between adjacent cross-sections using an iterative procedure called the standard step method, where the backwater equation is integrated as:

$$\left(d + z_0 + \frac{\alpha V^2}{2g}\right)_2 - \left(d + z_0 + \frac{\alpha V^2}{2g}\right)_1 = L\bar{S}_f + C\left|\frac{\alpha V_2^2}{2g} - \frac{\alpha V_1^2}{2g}\right| \qquad (6)$$

where the losses are separated into friction and contraction/expansion losses with $\bar{S}_f$ a representative friction slope between the two sections, L is the distance between the sections (which may be weighted, see Section 3.1), C is a contraction or expansion coefficient, d is the flow depth, z_0 is the invert level, and α is a velocity weighting coefficient.

The primary assumption of the integrated backwater equation used in steady-state numerical modelling is that the flow is gradually varied (Henderson 1966). This implies that changes along the channel, such as cross-section shape, invert level, flow depth and pressure distribution, are relatively small over short distances. The backwater equation has questionable or no accuracy in: (a) areas of rapid acceleration or deceleration, where the assumption of a hydrostatic pressure distribution is no longer valid, (b) areas of large turbulence and/or energy loss, and (c) areas of large change in cross-section property where the assumption that the representative friction slope and contraction/expansion losses can be estimated by some combination of the section properties at each end.

3.3. The Saint Venant Equation and Unsteady One-Dimensional Modelling

Unlike steady-state modelling, which uses a solution of the continuity and energy equations, unsteady modelling is based on a solution of the continuity and momentum equations. The derivation of these equations into a format suitable for one-dimensional modeling is complex but fairly well documented (e.g. Liggett 1975, Henderson 1966, reference manuals of HEC-RAS and MIKE 11). The vertically integrated equations of continuity and momentum, commonly known as the Saint Venant equations, may be presented as:

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} - q = 0 \quad \text{and} \quad \frac{\partial Q}{\partial t} + \frac{\partial QV}{\partial x} + gA\left(\frac{\partial z}{\partial x} + S_f\right) = 0 \qquad (7)$$

where Q is the discharge, q is the lateral inflow (per unit length), A is the flow area, z is the free-surface elevation and V is the velocity.

While the backwater equation is based on a steady-state differential form of the Energy equation, the Saint Venant equation based solutions can model unsteady flow conditions. Many software packages, including HEC-RAS (unsteady solver) and MIKE 11, adopt an algorithm that cannot accommodate two boundary conditions at the same boundary. As a consequence they cannot model supercritical flow, for which both discharge and water level are controlled by the upstream boundary. Instead, supercritical flow conditions are 'solved' by suppressing the convective acceleration as the Froude number increases. Earlier software such as RUBICON and ESTRY simply failed if supercritical conditions where encountered. The implications of this are discussed further in Section 6.2.

4. TWO-DIMENSIONAL MODELLING

Two-dimensional hydraulic models are commonly used for modelling of floodplains, coastal and marine situations where the flow path is poorly defined. Two-dimensional models calculate water depths and velocities across a grid or mesh that defines the topographic information. Traditionally, the mesh has been a fixed-space rectilinear grid with the governing equations solved using implicit finite-difference techniques. More recent models have allowed for a flexible mesh (typically consisting of triangles or quadrilaterals) solved using finite-element methods, which have significantly greater ability to handle complex geometries and boundaries at the expense of increased numerical complexity.

The numerical solution used by two-dimensional hydraulic models is usually based on the Saint Venant equations, which are derived from the depth-integrated conservation of mass and Navier-Stokes equations (Eq. (1) and (3) respectively). The Saint Venant equations are also commonly known as the shallow water equations, and are based on the assumption that the horizontal length scale is significantly greater than the vertical scale, implying that vertical velocities are negligible, vertical pressure gradients are hydrostatic, and horizontal pressure gradients are due to displacement of the free surface.

Unlike the algorithms used by one-dimensional models, two-dimensional models can often model both subcritical and supercritical flow conditions (see Section 6), although the user is advised to confirm this for any particular software package. For example, MIKE 21 by DHI Software requires at least two grid cells in the direction of flow to correctly resolve transition from sub- to supercritical flow at a control such as a weir (McCowan et al 2001).

In addition to the base assumptions of the Saint Venant equations discussed above, additional limitations are imposed by the formulations used to estimate the forces acting on each fluid component, such as viscous shear stresses and bed friction. Viscosity calculations in particular can be particularly vulnerable to 'water column' effects when the vertical length scale approaches or exceeds the horizontal scale. Figure 2(b) shows an example of the significant breakdown in shear stress calculations that can occur when the flow depth (5m) significantly exceeds the grid size (2m). While often based on a Manning's roughness, the implementation of bed friction within a two-dimensional model is different from a one-dimensional model. For example roughness is only included on the plan of the grid and not the walls, and the macro-scale effects of changes of channel shape and direction do not need to be accounted for by the roughness. It should be remembered that the Manning's roughness coefficient was developed for one-dimensional flow motion only.

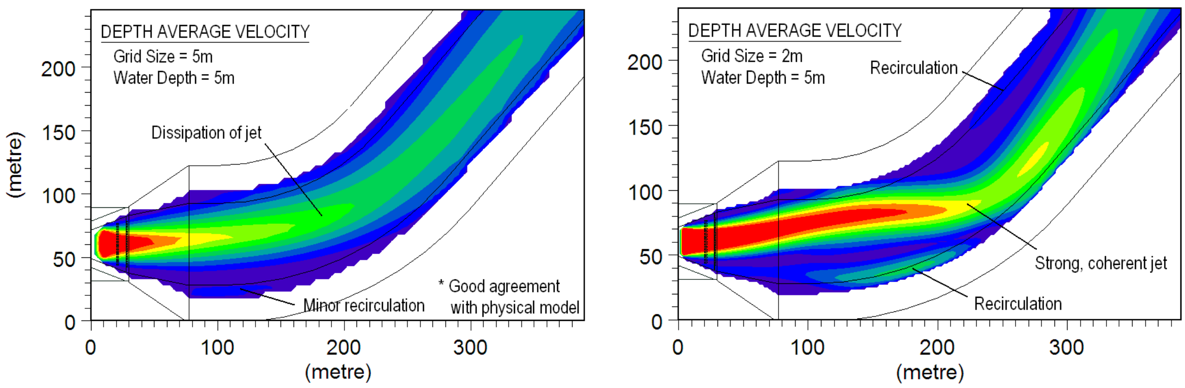

Figure 2 Velocity at a tunnel outlet with water depth ≈5m for (a) 5m grid, and (b) 2m grid

5. THREE-DIMENSIONAL AND HIGHER-ORDER MODELLING

Many of the limitations imposed by or on two-dimensional models are related to the assumptions they make in relation to depth, such as the hydrostatic pressure distribution or shear forces. Three-dimensional modelling should therefore theoretically remove many of these limitations, although it does so at the expense of increasing the complexity of the numerical computations by an order or several orders of magnitude. Models vary widely in terms of complexity and capability, from simply including another dimension into the Saint Venant equations to full "CFD Models" capable of modelling compressible fluids, multi-phase flow, thermodynamic effects and beyond. As such, it is difficult to generalise a specific set of limitations that apply to high-order models. It is nevertheless important to recognize that these limitations still exist regardless of how sophisticated the model may appear. Even with the great advancement in numerical modelling capability that has occurred in recent times, if the outcome is considered to be of importance or risk (e.g. dam spillways) it is still common practice to verify the numerical modelling results through the tried and tested means of physical modelling.

6. COMPARISON WITH PHYSICAL MODELLING

The capability and limitations of the various software packages listed in Table 1 were tested by comparing the model predictions with the results of two simple physical model experiments. The physical modelling was undertaken at the University of Queensland.

Table 1 – Summary of Software Tested Against Physical Models

Software	Version	Publisher	Category	Capability
HEC-RAS	4.0	United States Army Corps of Engineers	One-Dimensional	Steady ([a])
MIKE 11	2008	DHI Software	One-Dimensional	Unsteady
MIKE 21	2008	DHI Software	Two-Dimensional	Unsteady
FLOW-3D	9.2.1	Flow Science Inc.	3D CFD	Unsteady

Notes: a HEC-RAS has both steady-state (backwater equation) and unsteady (Saint Venant equation) modules. The results and general characteristics of the unsteady solver are similar to MIKE 11. For simplicity, only the MIKE 11 unsteady results are discussed, and may be assumed to also apply to HEC-RAS Unsteady.

6.1. Weir Experiment

The weir experiment consisted of a 3.2m long, 0.25m wide flat channel with a streamlined weir located approximately 1m from the upstream end (Figure 3). If the crest of the weir is 'broad' enough for the streamlines to become parallel to the weir crest, the pressure distribution will be approximately hydrostatic and the flow on the weir can be considered as gradually varied flow. If the upstream head is large compared to the crest length, the flow across the weir will be rapidly varied. Both conditions were examined on the physical model and compared with numerical model predictions. The objective of this comparison was to examine the capability of the software in the gradually- and rapidly-varying flow conditions that occurred along the weir. The model results are presented in Figure 3 and discussed below.

HEC-RAS: The backwater equation is not valid for rapidly varied flow. Nevertheless, HEC-RAS identifies that the weir acts as a hydraulic control and predicts that a transition from subcritical to supercritical flow occurs on the weir. For the low-flow case, HEC-RAS achieves a good match of the flow across the weir, with only minor differences occurring in the rapidly-varying flow at the upstream and downstream end. Significant differences are observed across the weir for the high-flow case, where the entire weir is rapidly varied flow. The profiles upstream and downstream of the weir are nevertheless predicted with good accuracy.
MIKE 11: Because of its numerical limitations, MIKE 11 cannot model the supercritical flow downstream of the weir. For the low-flow case, the downstream water level is over-estimated by a factor of 8 (consider the implications of an underestimate of velocity by a factor of 8!). This high tailwater impacts on the flow conditions on the weir, causing a significant error in the upstream water level. The incorrect tailwater has less impact for the high-flow case. There is still significant error in the predictions across the weir, but the upstream water level is almost correct. Can this prediction really be trusted, or is it just luck?

MIKE 21: The numerical algorithms used by MIKE 21 allow it to achieve a good match of the flow across the weir. Some differences are observed at the ends of the weir crest, particularly for the low flow case. This is believed to be partly numerical limitation and partly due to the difficulty of representing a smooth transition with a fixed space grid (i.e. a physical limitation).
FLOW-3D: With the ability to model vertical velocity components, flow acceleration and non-hydrostatic pressure, FLOW-3D achieves an excellent match of the physical model data.

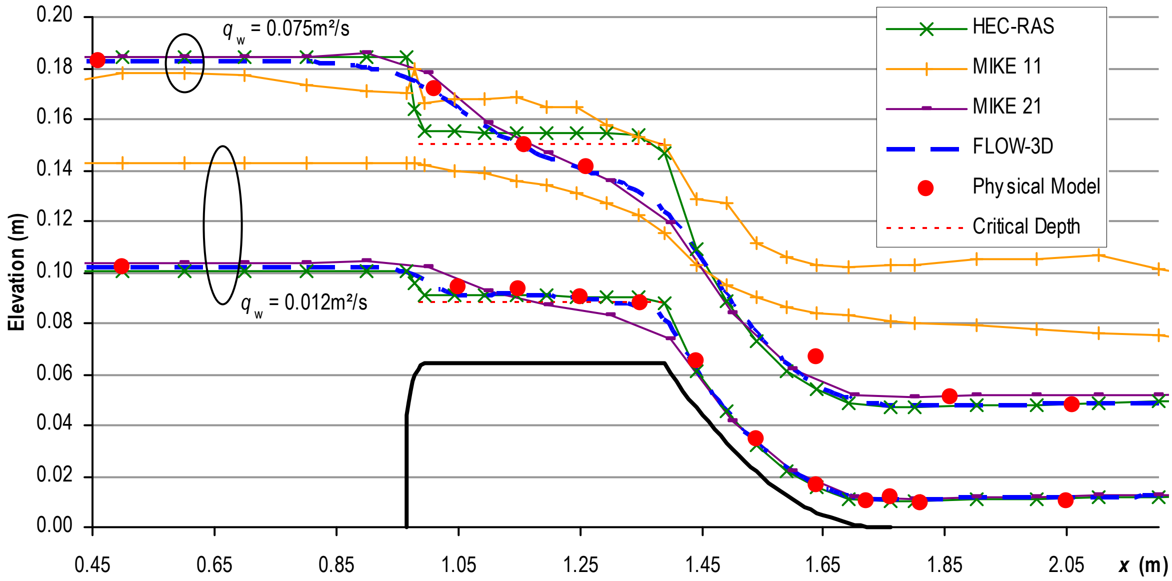

Figure 3 – Weir Water Level Predictions of the Physical and Numerical Models

6.2. Open Channel Flow Experiment

The Open Channel Flow experiment apparatus was a 12 m long, 0.5m wide rectangular flume set with a slope of 1.6º. Upstream and downstream water levels were controlled by a rounded gate, while constant inflow was provided by an upstream reservoir supplied from a constant-head tank. Key objectives of the physical model were to demonstrate: (a) the basic principles of open channel flow and the backwater equation, (b) the control mechanisms (i.e. sub-critical from downstream and super-critical from upstream), and (c) the transition from super-critical to sub-critical flow in the form of a hydraulic jump. Physical and numerical model results are shown in Figure 4 and discussed below.

HEC-RAS: The HEC-RAS (steady-state) model prediction shows good agreement with the physical model measurements. The steady-state solution of the Backwater Equation in HEC-RAS can model both subcritical and supercritical flow. HEC-RAS also identifies the occurrence of a hydraulic jump transition and predicts its location with reasonable accuracy. The jump is modelled as an instantaneous transition between cross-sections, and there is no attempt to describe the length or strength of the jump.
MIKE 11: As identified in Section 3.3, the solutions of the Saint Venant equation in both MIKE 11 and the unsteady solver of HEC-RAS are incapable of modelling supercritical flow. Although able to match the subcritical flow measurements, significant disparity is observed in both the supercritical flow region and hydraulic jump. While at first glance the result may appear conservative (i.e. predicted water levels higher than measured), this cannot be guaranteed in all circumstances. Additionally, when water levels are over-predicted, velocities will be under-predicted, which could have potentially catastrophic consequences.
MIKE 21: Although also based on the Saint Venant equation, the solution within MIKE 21 demonstrates the ability to model both subcritical and supercritical flow with good accuracy (although a supercritical upstream boundary cannot be modelled, and defaults to critical depth). The hydraulic jump is also modelled with reasonable accuracy. The model lacks the capability to model the complex flow patterns within the hydraulic jump, and predictions in this region should be treated with caution.
FLOW-3D: Considering the technological superiority of the FLOW-3D modelling package, it is just as well that the model prediction displays excellent agreement with the physical model. The software can also model additional flow properties such as turbulence intensity and air entrainment.

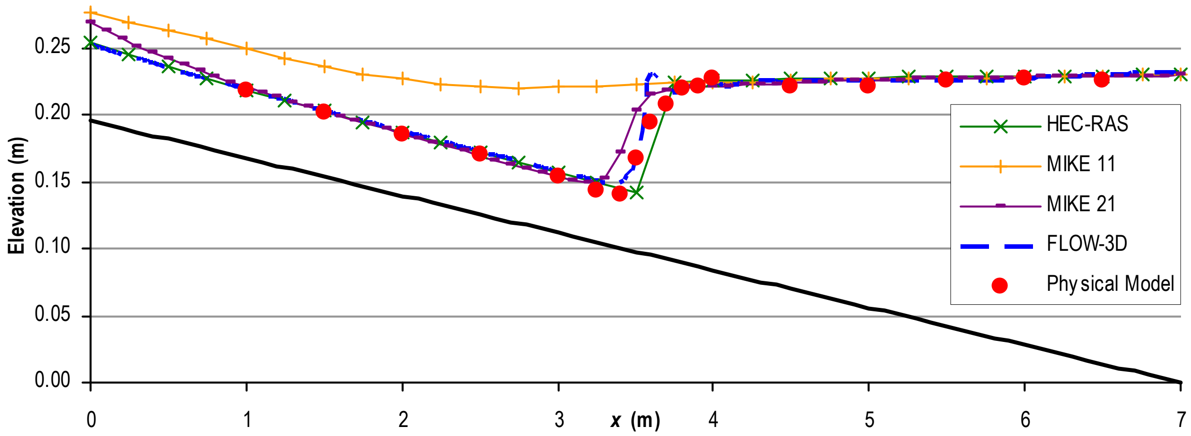

Figure 4 – Open Channel Flow Water Level Predictions of the Physical and Numerical Models

7. CONCLUSIONS

The study of hydraulics and fluid mechanics is founded on the three basic principles of conservation of mass, energy and momentum. Real-life situations are frequently too complex to solve without the aid of numerical models. There is a tendency among some engineers to discard the basic principles taught at university and blindly assume that the results produced by the model are correct. Regardless of the complexity of models and despite the claims of their developers, all numerical models are required to make approximations. These may be related to geometric limitations, numerical simplification (i.e. omission of 'unimportant' terms or fluid properties), or the use of empirical correlations. Some are obvious: one-dimensional models must average properties over the two-remaining directions, and two-dimensional models must assume depth-average flow properties. It is the less obvious and poorly advertised approximations that pose the greatest threat to the novice user. Some of these, such as the inability of one-dimensional unsteady models to simulate supercritical flow, or the 'water-column' effects of two-dimensional models, can cause significant inaccuracy in the model predictions.

A comparison of physical model and numerical model results confirmed that, if used in appropriate circumstances, numerical models can provide a good approximation of 'reality'. As soon as the software assumptions are violated however, the results of the model can no longer be trusted. In some cases the differences may be fairly minor, but in other cases the numerical model can significantly over- or under-predict water levels and velocities. The consequences of this are best left, and hopefully always left, to the imagination. Nevertheless, some advice can be provided to help avoid future problems. Firstly, it is recommended that anyone using any model should study carefully the documentation provided with the software, to fully appreciate what the model can do, and just as importantly, what it cannot do. Secondly, the user should realize that even the most sophisticated model is still required to make simplifications and use empirical approximations in its calculations. Great care should be taken when pushing the model beyond what has been tried and tested. Thirdly, even though sophisticated models can be applied to complex scenarios, it is important to remember that the basic principles of fluid mechanics still apply. It is often possible to perform a 'reality check' to verify that the model results are at least of the correct order of magnitude.

8. REFERENCES

Henderson, F.M. (1966). *Open Channel Flow*. MacMillan Company, New York, USA.
Liggett, J.A. (1975). *Unsteady Flow in Open Channels* WRP Publ. Vol. 1 Fort Collins, USA, pp. 29-62.
Liggett, J.A. (1994). Fluid Mechanics. McGraw-Hill, New York, USA.
McCowan A.D., Rasmussen E.B. and Berg P. (2001). *Improving the Performance of a Two-dimensional Hydraulic Model for Floodplain Applications*, Conference on Hydraulics in Civil Engineering, The Institution of Engineers, Hobart, Australia.
MIKE by DHI (2008) *MIKE 11 A Modelling System for Rivers and Channels Reference Manual*
USACE (2008). *HEC-RAS River Analysis System Hydraulic Reference Manual*, v4.0, California, USA